International Journal of Mechanical and Production Engineering Research and Development (IJMPERD)
ISSN (P): 2249–6890; ISSN (E): 2249–8001
Vol. 10, Issue 2, Apr 2020, 1419–1428


# ANALYSIS OF THE COOLANT FLOW IN SUB CHANNELS OF THE VVER-1000 REACTOR BY CFD METHOD

**SHOJAA AYED ALJASAR[1], OLEG.YU DOLMATOV[2], MUHAMMAD SAQIB[3] & YUBIN XU[4]**

*[1,2,4]School of Nuclear Science and Engineering, Tomsk Polytechnic University, Tomsk, Russia*

*[3]School of Energy Engineering, Tomsk Polytechnic University, Tomsk, Russia*

***ABSTRACT***

***Computational Fluid Dynamics (CFD) is a simulation tool that is used to analyse thermal phenomena and complex fluids. In this study, the coolant flow parameters in the sub-channels of fuel assemblies in the VVER-1000 reactor were examined by ANSYS R2. It was examined in different meshes accuracy and turbulence models to deal with the flow of water distribution problems such as velocity, pressure change as well as hydraulic resistance to the meshes of separation. The aim of this paper is to validate and design the VVER-1000 reactor taking into account turbulent flow heat transfer. First, the accuracy of different meshes and turbulence models is tested to deal with water flow problems such as velocity distribution, secondly the effect of axial distance and Reynolds number on the Nusselt number and pressure drop in the internal sub channel is studied numerically. A constant temperature field, pressure drop and velocity profile are illustrated to study and improve safety issues from an economic point of view. The results received are fully consistent with the measured values.***

***KEYWORDS: 3D Modelling, CFD, Fuel Assembly, Heat-Mass Transfer, Hydrodynamic, Sub Channel, Turbulent Flow, Nusselt Number, VVER & Pressurized Water Reactor***

**Received:** Mar 13, 2020; **Accepted:** Apr 03, 2020; **Published:** Apr 17, 2020; **Paper Id.:** IJMPERDAPR2020134

**Original Article**

## INTRODUCTION

The main application of a nuclear power reactor is the production of thermal energy that leads to power generation. A nuclear fission reaction in the fuel rod is the main source of heat. The heat energy produced in the fuel bar wall boundary transmits convection heat through the coolant. Turbulence and inconsistency in the flow area cause better heat transfer. Thus, turbulent flow heat transfer can be a major concern for forecasting and evaluating the thermal performance of power reactors. The thermodynamic properties in the VVER-1000 nuclear reactor were examined and analysed [1]. C.L Waata [2] studied heat extraction by coolant in the sub-channel of the VVER-1000 reactor. It was found that the heat transfer rate increased due to the increase in the axial distance. Markov et al. [3] Investigation of flow and heat transfer operations of the fuel collection model for the VVER-1000 reactor. It has been observed that the proposed model can be used to calculate heat transfer in reactor bar fuel elements. The various nuclear reactors may be of interest. But, the VVER-1000 nuclear reactor is a research choice. VVER-1000 is a Pressurized Water Reactor (PWR) that has a sprinkler suppression system. Hexagonal fuel bar assembly is the defining characteristic of VVERs. In nuclear reactors, heat is generated by the interaction of nuclear fission initiated by neutrons from any neutron source such as Am-Be. This heat is extracted by any coolant such as liquid sodium, heavy water, and light water. In VVER-1000, light water is used as a coolant. In PWRs, two cooling circuits are used to avoid radioactive contamination. The primary cooling circuit includes the sub-channels between the fuel rods and a heat exchanger immersed in the secondary circuit. Irradiated contaminated water is circulated in the

primary cooling circuit and never allowed out. The hexagonal fuel rod assembly contains 3 types of sub-channels according to the geometrical shape whose boundary conditions vary. These sub-channels are called the inner, angle, and edge sub-channels [4]. In this paper, only the results of the internal sub-channel are studied in the fuel rod group. Paseo et al. [5] He studied the thermal-hydraulic behaviour of the working fluid in a hexagonal bar bundle with a digitally mesh separator. It was found that transferring the disturbed momentum along two groups leads to increased heat transfer rate and results in better thermal performance. Rehme et al. [6] Reduced pressure in the penis bundles using the backflow experimentally. A link to stress loss due to the spacing network was provided and compared to the experimental data set. Jian et al. [7] Investigation of the analytical prediction of the friction factor and the number of Noselt close to the rod bundles. It was found that the method was valid for an infinite number of rod bundles. The aim of this paper is to validate and design the VVER-1000 reactor taking into account turbulent flow heat transfer.

## NUMERICAL SIMULATIONS ANALYSIS

### Numerical Model of the Sub-Channel

For the numerical simulations of fluid flow in fuel assembly, A central sub channel during a VVER-1000 fuel assembly that is enclosed by fuel rods solely was simulated. The simplicity of the model in shown in Figure 1.

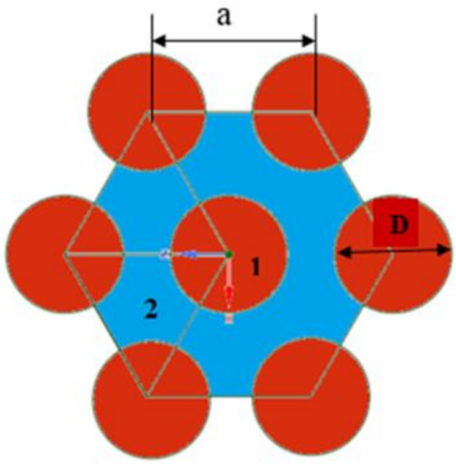



**Figure 1: Simplified Geometry of Part of Fuel Assembly: 1) Fuel Assembly 2) Coolant.**

The 62 mm long model of this sub-channel was designed with 9.1 mm fuel rod diameter (D) and 12.75 mm pitch (a) [8]. The boundary condition has been used for each flow field and thermal field. For flow field, the solid walls are assumed to be of no slip condition. For thermal field, the heat flux is specified at the fuel rod walls whereas the heat flux of 278.7 KW/m2 is used for modelling the domain. For the inlet, velocity and temperature are 5.5 m/s and 291° C respectively. The fluid flows upward within the sub-channel of the fuel rod assembly.

ANSYS R2 CFD code was used to build the geometrical model and to form different numerical meshes of the sub channel [10]. The geometry was resolved with four different meshes in Figure 2 to analyse the mesh resolution to the parameters. The questionable extruded mesh was used to discretise the model in space. The core region meshed with prism components; hexahedral cells were applied within the close to wall region. The cross-sectional resolution of the meshes was different in Figure 2, in the axial direction, a similar number of the layer was applied. The most characteristic of the meshes is often found in Table 1.

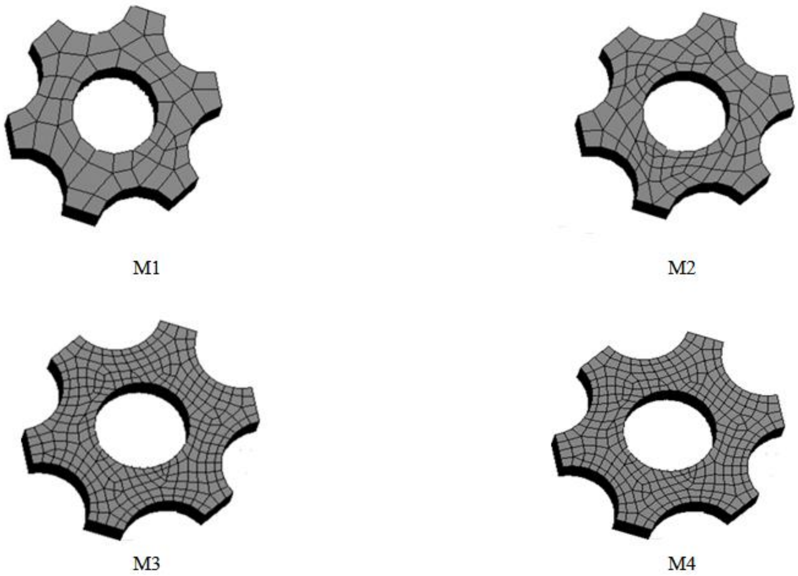



**Figure 2: The Different Mesh Resolutions of the Sub Channel.**

**Table 1: Characteristics of the Different Meshes**

| Mesh | M1 | M2 | M3 | M4 |
|---|---|---|---|---|
| Number of nodes | 3000 | 6100 | 25000 | 127700 |
| Number of elements | 2000 | 4600 | 20400 | 113250 |
| $y^+$ | 1720 | 1288 | 708 | 425 |

**Mesh Sensitivity Analysis Results**

The velocity contour and temperature distributions were affected directly from the different mesh resolutions, as shown in the Figure 3 and Figure 4. The streamlines of M1 and M2 were not symmetrical. In distinction, M3 and M4 streamlines were clear symmetry and nearer to real physics behaviour.

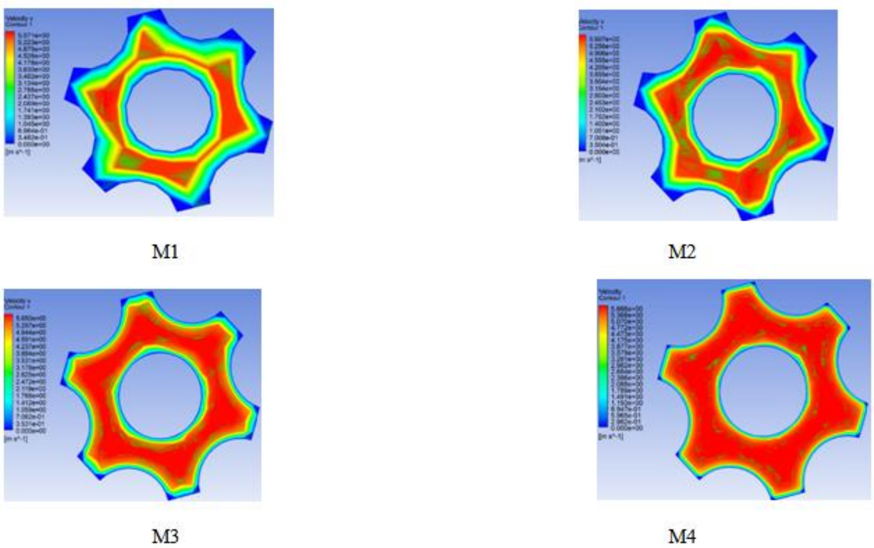



**Figure 3: Velocity Contours at Cross-Section in Case of Different Mesh Resolutions.**

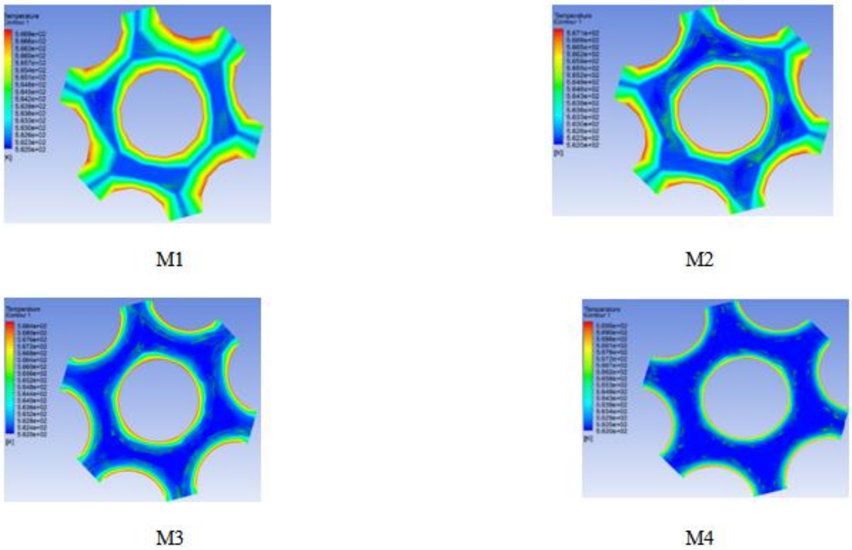
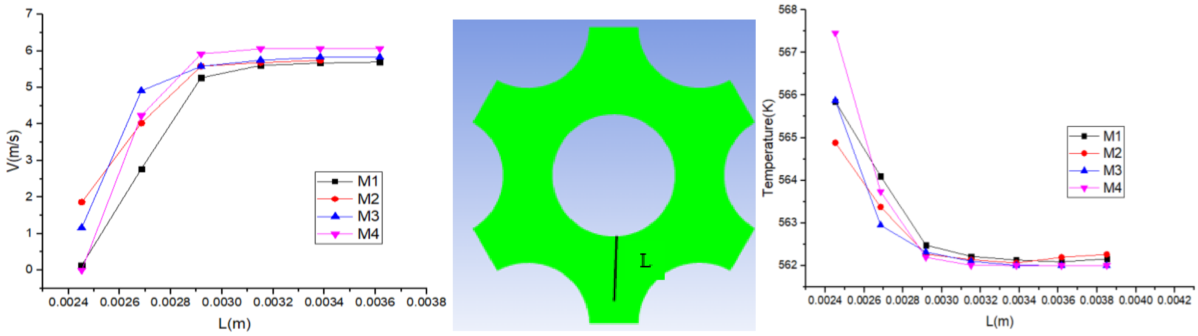



**Figure 4: Temperatures Distributions at Cross-Section in case of Different Mesh Resolutions.**

The results of the calculations were compared with each other along the "L" line using the provided coordinate system in Figure 5. It is illustrated that the best physical description for the axial velocity and temperature is M4.




**a) Axial Velocity with the along the "L" line b) Temperatures Distributions with the along the "L" line**

**Figure 5: Axial Velocity and Temperature with the different Meshing Types.**

**Results of the Comparative Study of Turbulence Model**

Different turbulence models were tested on M4 to check their impact on the axial velocity distribution and secondary flows. The k- ε, k-ω, SST and BSL Reynolds Stress turbulence models are chosen for the study. The basic assumption of k-ε model is that the dissipative impact of turbulence may be accounted for with a scalar isotropic property referred to as turbulent viscosity μt (Bousinnesq approximation). Turbulent viscosity is calculated regionally within the computational domain and is expounded to the native turbulent length and velocity scales. The assembly terms within the transport equation of turbulent quantities are associated with the native gradients of the mean flow. The advantages of k- ε model are relatively simple to implement, leads to stable calculations that convers relatively easily and reasonable predictions for many flows. The disadvantages of the k- ε model are poor predictions for swirling and rotating flows, flows with strong separation, axisymmetric jets, certain unconfined flows, and fully developed flows in non-circular ducts. [9, 10].

The k-ω model is highly correct from k-ε within the proximal layers of the wall, hence it has been successful in moderate-negative gradients of pressure, however it has failed in the flows with the pressure-induced separation [11]. In addition, the equation shows a high sensitivity of the ω values inside the free course outside the boundary layer [12]. In

most cases, the free current sensitivity deprived the equation of substituting the equation because the standard equation is in modelling disturbances, despite its superior performance within the region near the wall. This was the most motivating for this event from BSL and SST models.

BSL Reynolds pressure models belong to the second-order closure models in which the transport equations of individual Reynolds spaces are solved. BSL Reynolds Stress is the k- ω based disturbance model.

It is discovered that the k-ε, k- ω and SST models did not accurately mimic secondary flows. Reynolds' stress models better computes real physics phenomena. The axial velocity distribution is approximately the same in the case of all turbulence models in Figure 6 and Figure 7.

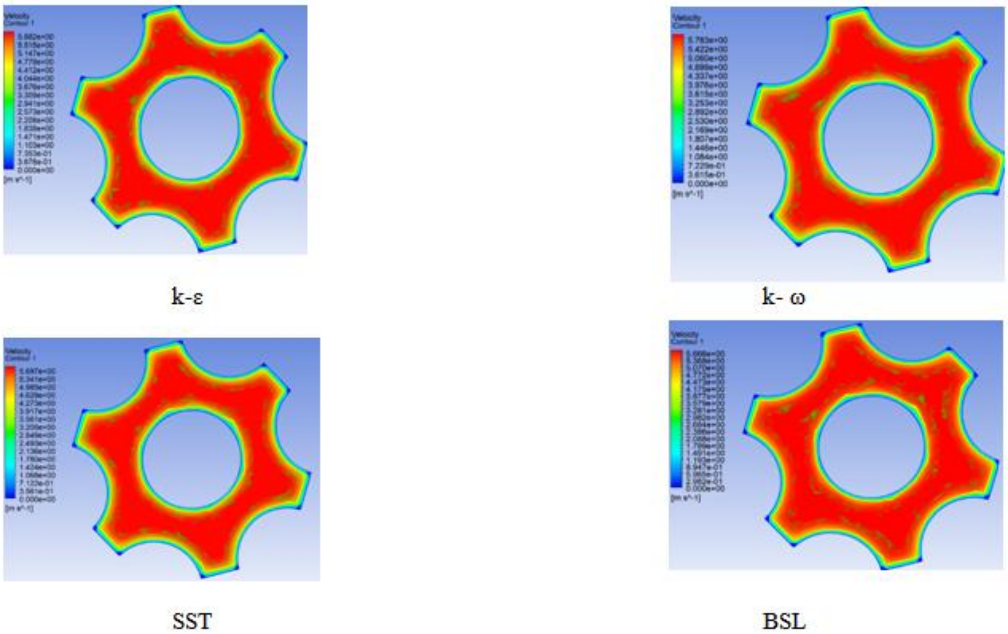



**Figure 6: Velocity Contours at Cross-Section in case of different Turbulence Models.**

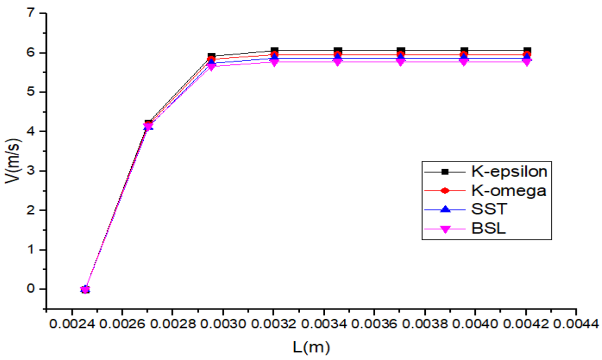



**Figure 7: Axial Velocities with the different Turbulence Models.**

**Simulation of 3D Coolant Flow in a Fuel Assembly**

Consistent numerical analysis of the performance of the entire VVER-1000 reactor running through an operational temporary pass requires advanced thermo couple hydraulic codes along with basic models of neutron 3D motion. Therefore, the ability to model embedded symbols allows to simulate both operational transfers and accident scenarios that take into account the evolution of the power distribution in the heat. Where, the energy distribution depends on the

interactions between the nucleus and the rest of the plant system. The CFD code to simulate mixing studies was ANSYS FLUENT [12]. Fluid Flow (Fluent) has the same functionality as the CFX module, but it contains a wider range of models and methods for modelling flows with chemical reactions. It also has a built-in grid editor. For numerical simulation of coolant flow in the fuel group, it is restricted in the engineering to have only one fuel rods.

The 3.53 m long model of this sub-channel was designed with 9.1 mm fuel rod diameter (D) and 12.75 mm pitch (a) [8]. The boundary condition has been used for each flow field and thermal field. For flow field, the solid walls are assumed to be of no slip condition. For thermal field, the heat flux is specified at the fuel rod walls, whereas the heat flux of 278.7 KW/m2 is used for modelling the domain. For the inlet, velocity and temperature are 5.5 m/s and 291° Crespectively. The fluid flows upward within the sub-channel of the fuel rod assembly. A transient simulation of the 3D coolant flow was performed. Figure 8 and Figure 9 shows contours in the sub-channel.

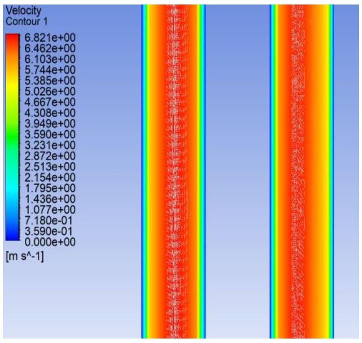



**Figure 8: Velocity Contours at Axial Cross-Section.**

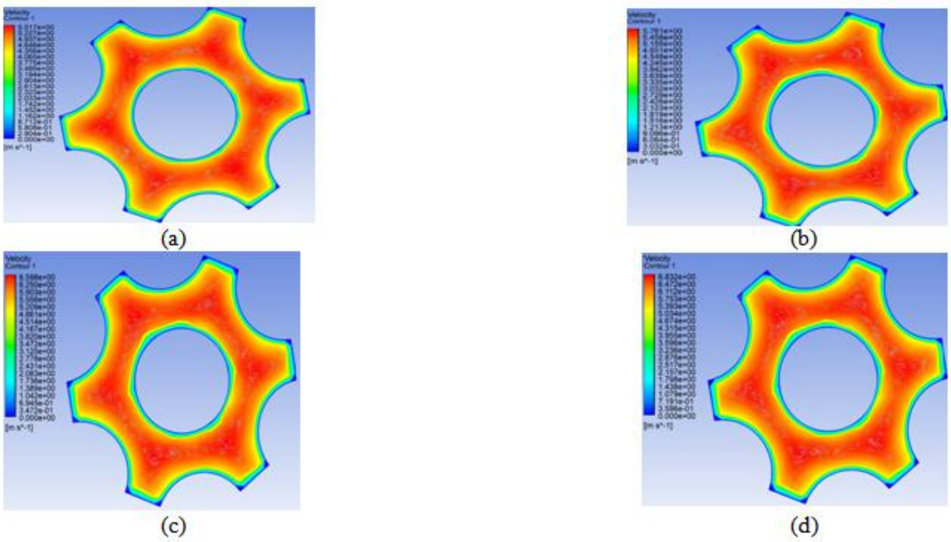



**Figure 9: Constant Velocity Contours in the Sub-Channel for (a) z = 0.25m, (b) z = 1.25m, (c) z = 2.75m, (d) z = 3.5m.**

Variation of cooling water velocity along the axial distance is shown in Figure 10. The increase of velocity at the end of the sub-channel is about 1.3 m/s. This increase in velocity compensates the loss of velocity due to the blockage in spacer grid.

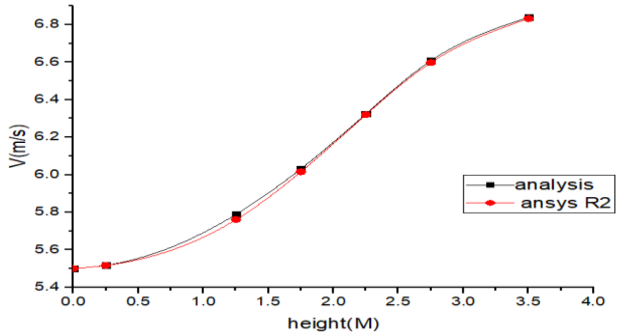



**Figure 10: Axial Velocity with the Height.**

In Figure 11, it is illustrated that the maximum pressure decreases with an increase in Reynolds number. It was also verified that for a higher number of Reynolds, the pressure gradient was lower.

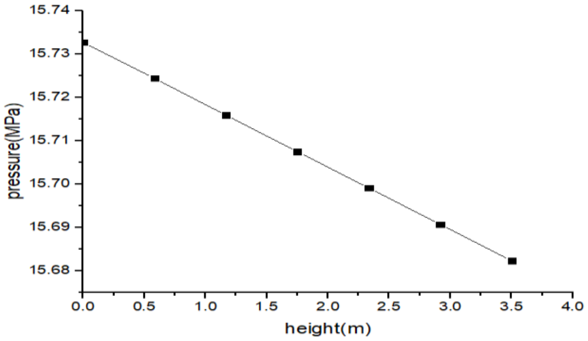



**Figure 11: Variation of Pressure with Axial Distance in the Sub Channel.**

In Figure 12, the variation of fully developed Nusselt number variance is shown in relation to Reynolds number. The increase in the number of Nusselt to get a higher Reynolds number is not very high. To operate the reactor economically, an improvement must be made between Reynolds number and the fully developed Nusselt number. A higher Reynolds number results in a higher operating cost, but an increased Nusselt number results in lower production costs per unit of energy.

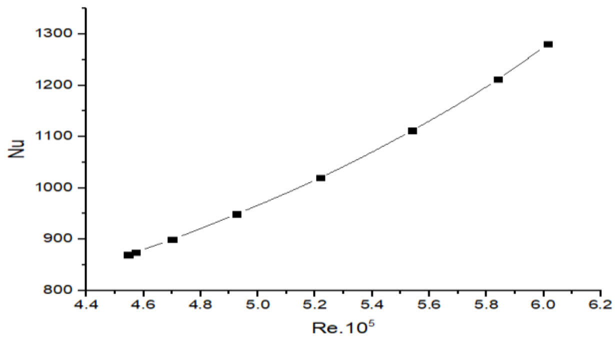



**Figure 12: Variation of Fully Developed Nusselt number with Reynolds number in the Sub-Channel.**

Engineers use friction factors (f) during this scheme to calculate the pressure drop and flow rates for flowing into smooth, rough tubes. It is desirable to have an inter-relationship of data covering the entire range of Reynolds number, from laminar flow, transient flow, and till it reaches the highest Reynolds number. To do this, Morrison (2013) developed a new data threading relationship for smooth tubes, which is clearer and almost easy in form. [13]:

$$f = \left[\frac{0.0076\left(\frac{3170}{\Re}\right)^{0.165}}{1+\left(\frac{3170}{\Re}\right)^{7.0}}\right] + \frac{16}{\Re}$$

When Reynolds number is lower, this equation becomes (16 / Re) and when Reynolds numbers rise (4000≤ Re ≤ 106), this equation follows the relationship of Prandtl, the smooth-pipe equivalent and source of the Colebrook equation. In this study, the friction factors for the Reynolds number of different height to equation are calculated using the maximum velocity of the water, as shown in the Figure 13.

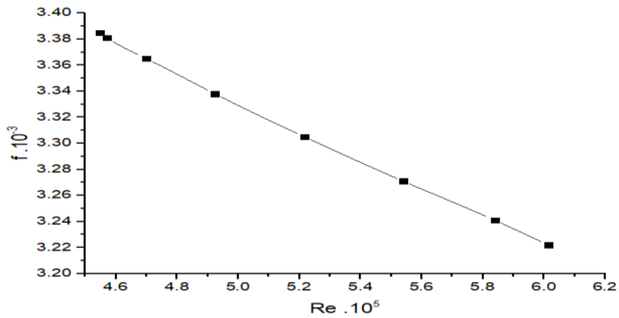



**Figure 13: Variation of Friction Factor with Reynolds Number.**

## CONCLUSIONS

The water flow properties of sub channel and rod bundle sections in VVER-1000 reactors were examined using the code ANSYS R2 Fluent. Sub-channel models were studied with some different mesh density. Sensitivity study showed that the appropriate network accuracy is very important to correctly predict the quantities of disturbances in a sub channel. In addition, the appropriate network has also been used to study disturbance models such as k- ε, k-ω, SST and BSL Reynolds Stress turbulence models. Based on these studies, the Reynolds BSL stress model was chosen for further investigations. CFD calculations of 3D coolant flow in a fuel assembly of the reactor VVER 1000 are performed with CFD commercial ANSYS Fluent tool, and velocity and pressure behaviours are examined. Good results were obtained from the simulation of 3D coolant flow in a fuel assembly of the reactor VVER 1000.

In the future, it is planned to investigate the full-dimensional fuel bundle model to provide detailed data on the properties of liquids and apply their results to the safety analysis and operation of nuclear power plants that will be built in Jordan.

## ACKNOWLEDGEMENTS

The authors express gratitude to Tomsk Polytechnic University, Russia for permitting to carry out this research work.

### *REFERENCES*

1. *A. A Falkov.,O. B Samoilov, (Ed).Experimental investigation and analysis of thermal-hydraulic characteristics of WWER-1000 alternative FA. 6th International Conference on WWER Fuel Performance, Modelling and Experimental Support. Albena, Bulgaria.*
2. *C. L Waata.(2005). Coupled Neutronics/Thermal-Hydraulics Analysis of a High-Performance Light-Water Reactor Fuel Assembly. FZKA, Forschungszentrums Karlsruhe, Karlsruhe, Germany 7233*

3. *P. V Markov.(2014) The numerical calculation of heat transfers from stem fuel elements of nuclear VVER. Science and education: science and technology electronic edition, 790-799.*

4. *Y. K Pandey, A Chauhan. Fuel Management of VVER-1000 Reactor of Kudankulam. Nuclear Power Plant, India, IAEA.*

5. *J Pacio., K Litfin.,(Ed). (2015). Heat transfer to liquid metals in a hexagonal rod bundle with grid spacers: Experimental and simulation results. (pp. 27-39). Nucl. Engg, Design 290.*

6. *Rehme K., Trippe G.(1980) Pressure drop and velocity distribution in rod bundles with spacer grids.(pp. 349-359). Nucl. Eng. Design, 62.*

7. *Jian S., Atila P., Silva F.(2002) Analytical prediction of friction factors and Nusselt numbers of turbulent forced convection in rod bundles with smooth and rough surfaces.(pp.111-127). Nucl. Engg, Des. 215.*

8. *IAEA- WWER-1000 Reactor Simulator, Material for Training Courses and Workshops, Second Edition, 2005*

9. *ANSYS Inc, Turbulence modelling.*

10. *ANSYS, Inc. ANSYS ICEM CFD Help Manual, 2012*

11. *F.R Menter.(1993). Zonal two-equation k-ω turbulence model for aerodynamic flows. AIAA Paper 1993-2906*

12. *F.R Menter.(1992) Influence of freestream values on k-ω turbulence model predictions. AIAA Journal, 30.6.*

13. *F.A Morrison.(2013). Data correlation for friction factor in smooth pipes. Department of Chemical Engineering, Michigan Technological University, Houghton, MI 49931*